International Journal of Engineering and Applied Sciences (IJEAS)
ISSN: 2394-3661, Volume-3, Issue-6, June 2016

Shape Effects of Wind Induced Response on Tall Buildings Using CFD

Sanket Lohade, Dr. Sanjay Kulkarni

Abstract- **High rise Structures are in demand due to scarcity of land in urban areas, economic growth, technological advancement, etc. Wind effect is very important for high rise structures and provides significant contribution to overall loading and serviceability. In order to provide design load and predict dynamic response, wind tunnel testing is an essential component in designing high rise building. As the wind tunnel testing is generally very expensive and time consuming, the use of CFD as an alternative to wind tunnel testing is studied. Different RCC buildings of varying shapes and aspect ratios are studied. An attempt is made to study the effect of different geometric configurations like flared, swastika, circular and tetragon of tall buildings having same plan area on Force coefficient. To study the wind effect, overturning moments, drag forces and torsional moments at base of the structures using Computational Fluid Dynamics(CFD) that is nothing but ANSYS FLUENT workbench. Then the numerical computation has been executed to validate the results of the same.**

Index Terms- **CFD, wind effect, overturning moments, drag forces, torsional moments**

I. INTRODUCTION

Earlier in 19^{th} century, there were no structures as tall buildings but with the technological advancement and rise in urbanization, there was a need for vertical expansion of cities. During that era, for design purpose, only vertical/gravity loads on buildings were to be considered but with increase in slenderness/height of buildings, lateral loads on structures i.e. wind loads and earthquake loads came into pictures which are more predominant. In 1930s' many high rise buildings were constructed in USA. It was a period of great prosperity for high rise buildings as extensive research work was carried out on wind induced effects on high rise buildings. Empire State building which was constructed in 1931 was the world's tallest building for next 40 years, used to vibrate like times of tuning fork without damaging the integrity of the structure but it caused discomfort to the occupants.

Unlike the mean flow of wind, which can be considered as static, wind loads associated with gustiness or turbulence rapidly and even abruptly, creating effects much larger than if the same loads were applied gradually. Wind loads, therefore,need to be studied as if they were dynamic in nature. This thesis will mainly concentrate on wind induced pressures which would be arises due to wind intensity and how pressure varies according to different shapes of buildings. The intensity of a wind load depends on how fast it varies and also on the response of structure. Therefore, whether the pressures on a building created by a wind gust, which may first increase and then decrease, are considered as dynamic or static depends to a large extent on the dynamic response of the structure to which it is applied.

Sanket vinod lohade,Civil engineering department, Dr. D.Y.Patil school of engineering and technology, Pune, India, 8087007222.

Dr Sanjay K Kulkarni, Head of Civil Engineering Department, Dr. D.Y.Patil school of engineering and technology, Pune, India, 9822953306

II. METHODOLOGY

For this study, Total 7 shapes of buildings according to plan has been selected which has been modelled in AutoCAD 3D and commercial CFD software Ansys Fluent 14.5 has been used to simulate full scale flow around building. CFD codes work by solving the governing equations with the use of a turbulence model. Different CFD commercial codes have different discretization methods to solve those governing equations. Fluent code uses finite volume discretization method to solve the governing equations, that means the region of interest (the domain) is divided into a finite number of cells or control volumes (the mesh or grid). In the simulation the variables are solved at the centre of the cell. The values at other locations are determined by using interpolating those values. In other words, the accuracy of numerical solution will usually improve with an increased number of grid points, especially if the increase is made in spatial regions with complex geometries. For this reason the creation of the mesh (or grid) is one of the most important issues too consider for a successful CFD simulation.

A. Size of computational domain:

There are no explicit rules for size of domain. The extent of the building area (e.g. surrounding buildings) that is represented in the computational domain depends on the influence of those features on the region of interest. As experience from wind tunnel simulations is that a building with height Hx may have a minimal influence if its distance from the region of interest is greater than 6-10 Hx. Thus, as a minimum requirement, a building of height Hx should be represented if its distance from the studied building complex is less than 6Hx. For this study, domain has been considered 800X600X400 in X, Y and Z direction respectively.

B. Shape Geometries:

General shapes have been selected and modelled using AutoCAD 3D. Swastika, Tetragon, Star, Flared, Trapezium, Kite, and Circle have been modelled and analysed further. Fig 1 shows circle geometry and in similar way other shape geometries have been modelled. Plan area and height has been kept same for all the shapes i.e.1250 m^2 and 110 m respectively.

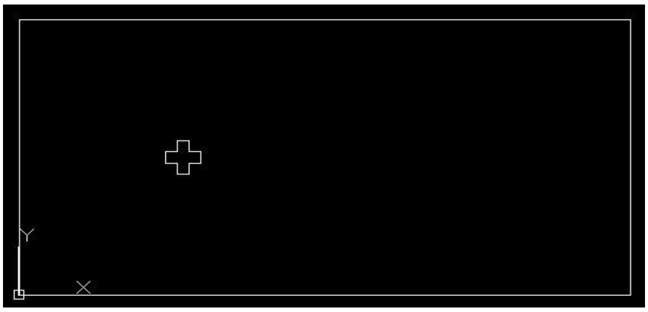


Fig 1 Swastika Shape geometry with Domain

C. Properties of materials:

1. Fluid:
Material around building**: Air**
Temperature : (300 k)
Density (kg/m3): 1.1777
Specific Heat Capacity: (Cp) – J/kg-K 1005
Thermal Conductivity: (W/m-K) - 0.0262
Kinematic Viscosity: (kg/m-s) - 1.5761e-05

2. Solid (Building)
Material: RCC
Density: 2500 kg/m3
Elastic Modulus: 28000 MPa
Poisson's ratio: 0.2

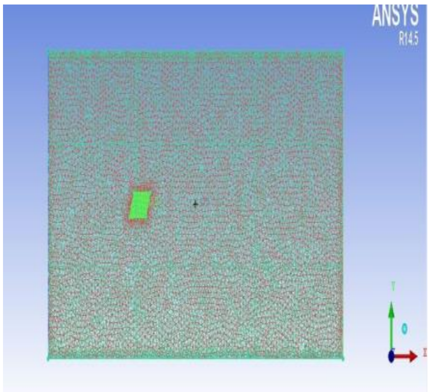
D. Mesh Generation:

To achieve the desired mesh i.e. with aspect ratio around 10 and orthogonal quality of 0.4, ICEM Mesh tool has been used. Tetragonal unstructured type of mesh has been performed with the following element sizes for particular region:

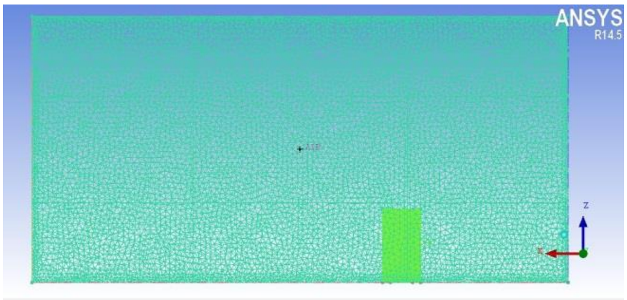
TABLE 1
SIZE OF ELEMENTS GIVEN IN ICEM MESH

Parts	Size of element
Inlet	10
Outlet	10
Walls	15
Building	1 with prism layers
Bottom	5
Inner curves	1
Outer curves	1

Fig 2 Shows The Complete Mesh For All The Parts As Follows:



Plan



Elevation
Fig 2 Mesh Generation

III. ANALYSIS AND RESULTS

Analyses have been performed in Ansys Fluent. For analyses, K-€ Realizable turbulence model with standard wall function has been used and Boundary conditions as follows:

- Inlet (velocity inlet) = 39 m/s
- Outlet = Outflow
- Lateral sides, top side and ground: wall
- Building Wall: Wall

After running the calculations till the convergence, results have been checked in CFD Post. Results consists of variation of dynamic pressure, pressure coefficients and drag force on tall building surface due to variation of geometric plan shape of tall building. Dynamic wind force by gust factor method has also been calculated according to IS875: Part 3 and compared with CFD drag force. The plan area for each building geometry is constant i.e.1250 square meter and height of each building geometry is 110 m. The value of force coefficient for each tall building plan shape geometry as well as the total drag force acting on the tall building unit as obtained by CFD code fluent is tabulated in table 1and 2 and also by graphically.

TABLE 2
COMPARISON OF GUST LOAD AND CFD DRAG LOAD

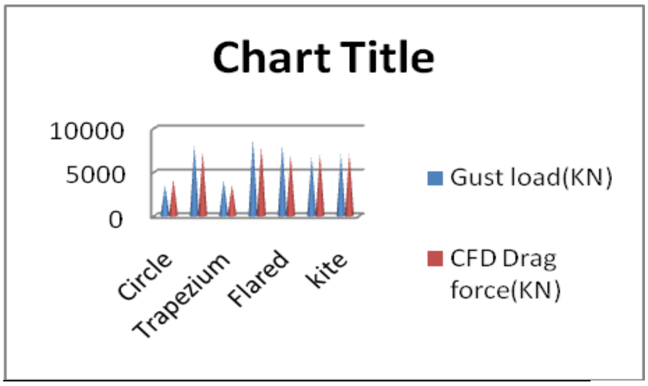
Geometry	Gust load(KN)	CFD Drag force(KN)	% Difference w.r.t CFD
Circle	3459.7	4179.08	17.21383654
Star	8133	7241	-10.9677
Trapezium	4038.8	3532.98	-14.31709209
Swastika	8807	7942	-9.82173
Flared	8120	6986	-13.25
Tetragon	6700.02	7018.54	4.538265793
kite	7048.4	7164	1.613623674

TABLE 2
VALUES OF DYNAMIC PRESSURE AND COEFFICIENTS

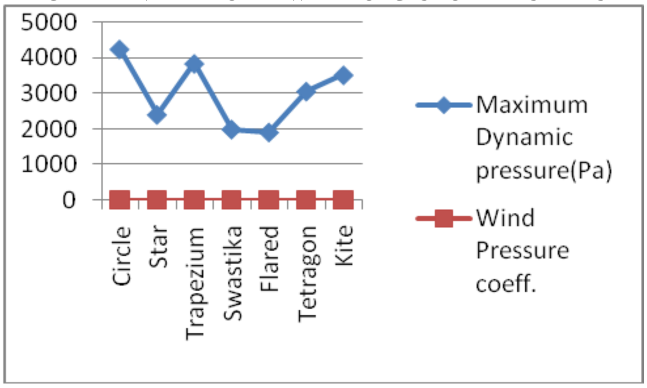
Geometry	Maximum Dynamic pressure(Pa)	Wind Pressure coeff.	Skin friction coeff.
Circle	4248.34	0.99	3.54E-03
Star	2409	1.09	3.90E-03
Trapezium	3843.89	1	9.50E-03
Swastika	1997	1.14	3.30E-03
Flared	1914	0.835	2.80E-03
Tetragon	3063.74	1.08	6.25E-03
Kite	3521.49	1.07	8.90E-03

ISSN: 2394-3661, Volume-3, Issue-6, June 2016

After comparing the forces it has been observed that trapezium shape having very less drag force and gust load. It is higher in case of swastika shape due to more angularity in edges of swastika hence it possesses high galloping effect near the edges near to the inlet and hence has higher drag force. Variation has been shown graphically as follows:



GRAPH 1 VARIATION IN WIND FORCES FOR ALL SHAPES



GRAPH 2 VARIATION IN DYNAMIC PRESSURE (FROM CFD)

A. Wind flow profiles of different plan shapes of buildings:

CFD is an effective tool to study the behaviour of wind on buildings. We see different shapes of buildings in practice. Following fig indicates how wind flows across the buildings

For circular building we can see the maximum effect of wind occurs on the sides of building which creates maximum gusts on side areas only. Vortex shredding is predominant on side edges of circular building as shown below.

It is to note that orientation of building is normal to the inlet. According to literature survey, circular buildings are most efficient in reducing wind load moments. It's obvious and from results obtained in this study, Circular buildings have less gust load and drag force because of smooth surfaces which creates less vortex shredding and less friction and hence more efficient in reducing moments due to wind excitation.

For tetragon it has very less facade area but edges are large which results in highest galloping effect across edges as shown in fig. we can observe the reverse oscillations induced due to galloping effect.

Flared buildings which have two edges at rear are responsible for maximum pressure induced as shown in fig. We can clearly observe transverse oscillations for flared buildings.

An edgy building like trapezium possesses almost same wind profile. Very less vortex shredding and galloping occurs in trapezium buildings as shown below

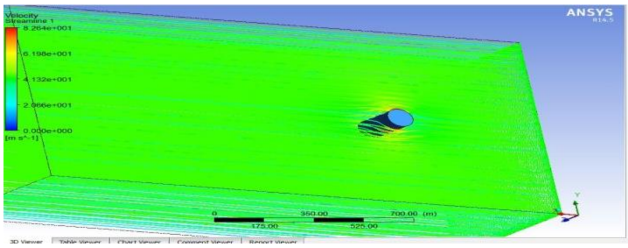


FIG 3.WIND FLOW PROFILE FOR CIRCULAR BUILDING

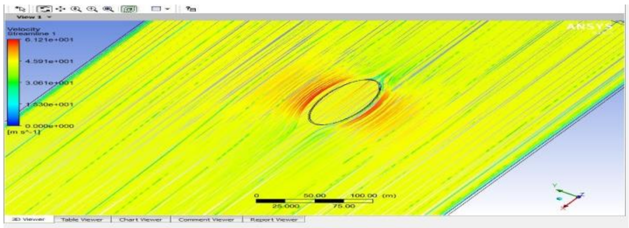


FIG 4.WIND FLOW PROFILE FOR ELLIPSE BUILDING

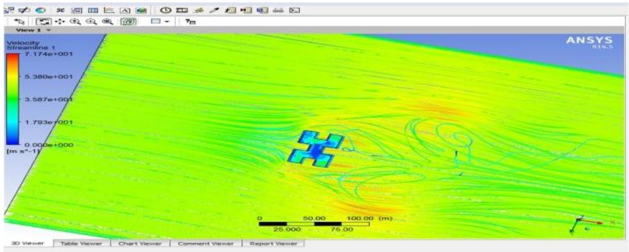


FIG 4.WIND FLOW PROFILE FOR SWASTIKA BUILDING

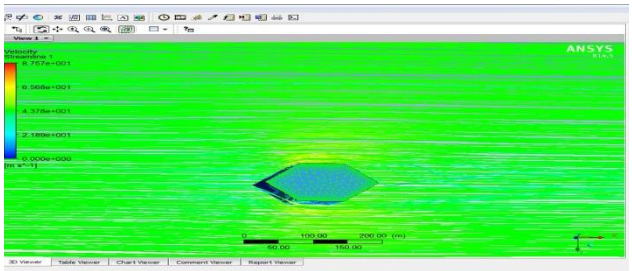


FIG 5.WIND FLOW PROFILE FOR TRAPEZIUM BUILDING

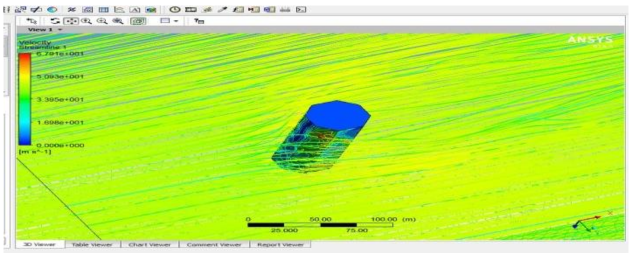


FIG 6.WIND FLOW PROFILE FOR TETRAGON BUILDING

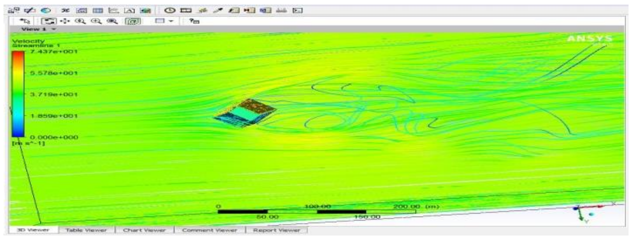


FIG 7.WIND FLOW PROFILE FOR INVERTED E SHAPE BUILDING

IV. CONCLUSION

CFD has been proved to be very effective tool for wind load analyses. From the results obtained, it was found that Wind Forces were maximum in kite and tetragon and were minimum in case of circle and trapezium shape of tall

building. The flared plan shape of tall building is more effective in reducing wind pressure coefficient than circular plan shape of tall building. Trapezium and Tetragon are more effective in reducing wind pressure coefficients than Triangle which has highest wind pressure coefficients. Drag force is very less for Trapezium. Also in case of max dynamic pressure it is very high for Circle shape. In brief, the circular and swastika plan shape of buildings is much better compared to the other plan shape of building in reducing both Wind Pressure Coefficient as well as Total Drag Force on Building. From velocity profiles obtained it was observed that vortex shredding and galloping effects are predominant in case of trapezium and flared. As edginess increases more transverse oscillations tend to induce. This results in more bending moment on building due to wind.

It is also noted that, orientation of building plays an important role in reducing gust load and moment.

ACKNOWLEDGEMENTS

I express my profound gratitude to our project guide Dr. Sanjay K. Kulkarni for their inspiring guidance due to which our difficulties and questions were shaped into the development of this project and complete support, co-operation and valuable suggestions.

I would like to thank Principal, Dr. Ashok S. Kasnale and H.O.D, Dr. Sanjay K. Kulkarni of Civil Department, who were the source of inspiration throughout the making of this project stage I and helped us to accomplish our goals in a much easier and healthy.

REFERENCES

[1] J.A.Amin and A.K.Ahuja, Effects of Side Ratio on Wind Induced Pressure Distribution on Rectangular Buildings; Hindawi Publishing Corporation Journal of Structures.

[2] Anupam Rjjmani, Analysis of Wind and Earthquake Load for Different Shapes of High Rise building; International Journal of Civil Engineering and Technology.

[3] An explanatory handbook on "Indian Standard code practice for design loads",(other than earthquake) for buildings and structures part 3 wind loads[IS 875 (part 3):1987]",Bureau of Indian Standards, New Delhi.

[4] Er.Ankit Mahajan, Er. Puneet Sharma, Er.Ismit Patel Singh,Wind effects on isolated buildings with different sizes through CFD simulation, IOSR journal of mechanical and civil engineering(IOSR-JMCE), volume 11, Issue 3 VER.4 (May – June 2014).

[5] Y.Tamura, Wind and Tall Buildings;EACWE 5, Florence,Italy; 19-23 july 2009.

[6] Ahsan Kareem, Seymour Spence, Using Computational Fluid dynamics to optimize Tall Building Design, CTBUH journal, Issue 3,2013.

[7] Ehsan Vafaeihosseini, Azadeh Sagheb, Analysis of Highrise Building using Computational Fluid Dynamics Approach: A case study on 38-storey Highrise Building, Structural Engineering World congress, April 2011.

[8] Morimasa Watakade, Comparison of wind pressure Measurement on Tower like structure obtained from Full scale Observation, Wind Tunnel Test, and CFD technology, published in journal of Wind Engineering and Industrial Aerodynamics, 2002:1817-1829.

[9] Qing Shan Yang, Simulation of horizontally homogeneous Atmospheric Boundary Layer based on K-ε Varied Model Combined with Modified Wall Functions, APCWE-vii, The Seven Asis-Pacific conference on Wind Engineering, November 2009.